



**US Army Corps
of Engineers®**
Engineer Research and
Development Center

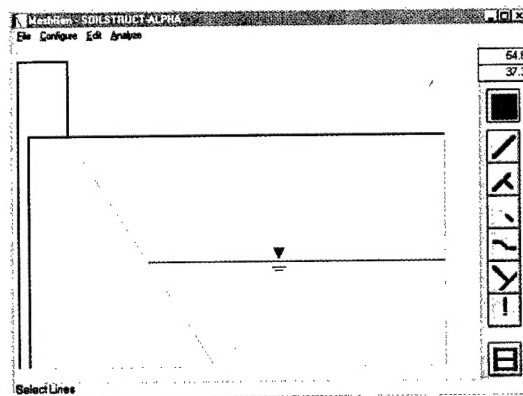
Computer-Aided Structural Engineering Project

SOILSTRUCT-ALPHA for Personal Computers

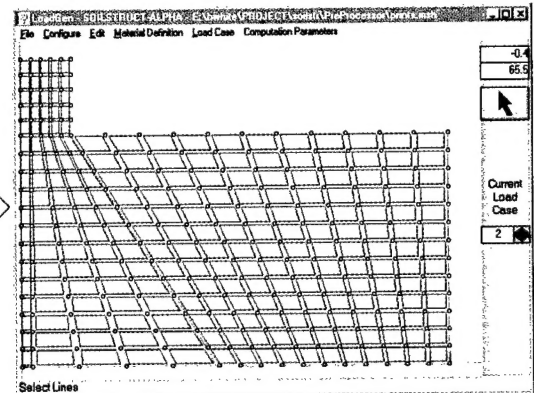
Report 1: Visual Modeler

Barry C. White and Robert M. Ebeling

December 2001



MeshGen



LoadGen

20011031 161

The contents of this report are not to be used for advertising, publication, or promotional purposes. Citation of trade names does not constitute an official endorsement or approval of the use of such commercial products.

The findings of this report are not to be construed as an official Department of the Army position, unless so designated by other authorized documents.



PRINTED ON RECYCLED PAPER

SOILSTRUCT-ALPHA for Personal Computers

Report 1: Visual Modeler

by **Barry C. White**

Computer Systems Technology, Inc.
3530 Manor Drive, Suite 5
Vicksburg, MS 39180

Robert M. Ebeling

Information Technology Laboratory
U.S. Army Engineer Research and Development Center
3909 Halls Ferry Road
Vicksburg, MS 39180-6199

Report 1 of a series

Approved for public release; distribution is unlimited

Prepared for **U.S. Army Corps of Engineers**
Washington, DC 20314-1000

and **Work Unit 31589**

Contents

Preface	v
1—Introduction.....	1
1.1 Overview of SOILSTRUCT-ALPHA Finite Element Program.....	1
1.1.1 Visual Modeler.....	3
1.1.2 SOILSTRUCT-ALPHA Finite Element Program.....	5
1.1.3 Visual Postprocessor	5
1.2 Modeling Guidelines	5
1.3 Hyperbolic Soil Modeling Parameter Guidelines	7
1.4 Interface Modeling Parameters Guidelines.....	8
1.5 Example Structure	10
1.6 Nomenclature	11
2— <i>MeshGen</i> , the Mesh Generator.....	12
2.1 Step 1: Initialize the Editor for the Model	12
2.2 Step 2: Define the Boundaries of the Model.....	13
2.3 Step 3: Break the Material Boundaries into Four- or Three-sided Regions.....	13
2.4 Step 4: Mesh the Regions	15
2.5 Step 5: Assign Interface Lines.....	16
2.6 Step 6: Generate Mesh (which will be input into the Load Generator).....	17
2.7 Step 7: Optimize Mesh (optional).....	17
2.8 Step 8: Verify Mesh – Node Numbers and Element Numbers	18
2.9 Step 9: Refine and Reiterate	19
3— <i>LoadGen</i> , the Load Generator.....	20
3.1 Step 1: Load the Mesh created in <i>MeshGen</i>	21
3.2 Step 2: Assign Initial Material Properties for 2-D and Interface Elements	22
3.3 Step 3: Assign Fixities for the Model	23
3.4 Step 4: Assign Initial Materials to the Mesh.....	23
3.5 Step 5: Create Initial Bar Elements (Optional)	24
3.6 Step 6: Create Load Cases	24
3.7 Step 7: Add Regions and Conditions for Each Load Case (Such as Fill Regions, Uplift Conditions, etc.).....	25

3.8 Step 8: Create Spreadsheet Output (Optional).....	26
3.9 Step 9: Set Parameters for Analysis (Optional)	27
3.10 Step 10: Set Restart Information (Optional)	27
3.11 Step 11: Save the Loads into an .lds File.....	28
3.12 Step 12: Create SOILSTRUCT-ALPHA Input File	28
4—Summary	29
References.....	31
SF 298	

Preface

The work described herein was sponsored by Headquarters, U.S. Army Corps of Engineers (HQUSACE), as part of the Civil Works Research and Development Program on Structural Engineering (CWR&D). The research was conducted under Civil Works Work Unit 31589, the "Computer-Aided Structural Engineering (CASE)" Project for which Dr. Stanley C. Woodson, Geosciences and Structures Laboratory, Vicksburg, MS, U.S. Army Engineer Research and Development Center (ERDC), is Problem Area Leader and Mr. H. Wayne Jones, Computer-Aided Engineering Division (CAED), Information Technology Laboratory (ITL), Vicksburg, MS, ERDC, is the Principal Investigator. The HQUSACE Technical Monitor is Mr. Jerry Foster, CECW-ED.

The SOILSTRUCT-ALPHA for Personal Computers analysis package comprises three separate stages: the Visual Modeler, the SOILSTRUCT-ALPHA Finite Element Analysis Program, and the Visual Postprocessor. This report, Report 1 of a series, describes how the SOILSTRUCT-ALPHA Visual Modeler (i.e., a preprocessor) is used, applying it to the suggested work flow of a typical project as an example. This report is intended to serve as a user's guide and not as a reference manual defining each specific command. Report 2 will describe the Finite Element Analysis Program, and Report 3, the Visual Postprocessor.

The work was performed at ITL by Mr. Barry C. White, Computer Systems Technology, Vicksburg, MS, and Dr. Robert M. Ebeling, CAED. Dr. Ebeling was the author of the scope of work for this research. The report was prepared by Mr. White and Dr. Ebeling under the direct supervision of Mr. Jones and Mr. Timothy D. Ables, Acting Director, ITL.

At the time of publication of this report, Dr. James R. Houston was Director of ERDC, and COL John W. Morris III, EN, was Commander and Executive Director.

The contents of this report are not to be used for advertising, publication or promotional purposes. Citation of trade names does not constitute an official endorsement or approval of the use of such commercial products.

1 Introduction

The SOILSTRUCT-ALPHA for Personal Computers analysis package comprises three separate stages: the Visual Modeler, the SOILSTRUCT-ALPHA Finite Element Analysis Program, and the Visual Postprocessor. This report, Report 1 of a series, describes how the SOILSTRUCT-ALPHA Visual Modeler (i.e., a preprocessor) is used, applying it to the suggested work flow of a typical project as an example. This report is intended to serve as a user's guide and not as a reference manual defining each specific command. Report 2 will describe the Finite Element Analysis Program, and Report 3, the Visual Postprocessor.

1.1 Overview of SOILSTRUCT-ALPHA Finite Element Program

Engineering application of soil-structure interaction modeling requires a balance between modeling realism and simplicity. Construction procedures have been modeled by the finite element method for over 25 years, and the key ingredients in engineering applications are well known. One of the earliest successful applications of soil-structure interaction analysis was performed by Clough and Duncan (1969; Duncan and Clough 1971) in their analysis of the two reinforced concrete U-frame locks at Port Allen and Old River. These two locks had been extensively instrumented. Prior to Clough and Duncan's analyses, the instrumentation data had been thought to be unreliable and contrary to the perceived understanding of the behavior of locks encountered during lock operation. Clough and Duncan's study showed that the best agreement between results computed using the finite element method and those obtained through instrumentation measurements is achieved when the actual construction process is simulated as closely as possible in the analysis. During their study, Clough and Duncan developed what is referred to as the backfill placement analysis in which the loads exerted by the backfill on the lock wall are generated automatically during simulated placement of backfill behind the wall (i.e., predetermined earth pressure force distributions between the soil and the lock are not specified). This requires that the soil backfill and foundation soil strata be included in the finite element mesh. This procedure involved the use of incremental finite element analysis with nonlinear, stress-dependent, stress-strain behavior for the soil. An additional requirement is that interface elements be incorporated within the finite element mesh to allow for relative movement between the soil and structure. Clough and Duncan developed the first version of the finite element program SOILSTRUCT that implements this engineering modeling/analysis philosophy.

SOILSTRUCT has been used successfully in the past decade on several engineering projects supported by field observations. Since 1969 several versions of SOILSTRUCT have been developed to analyze various types of earth retaining structures or to analyze specific problems that were not envisioned at the time of Clough and Duncan's original development. One of these updated versions is referred to as SOILSTRUCT-ALPHA and is the subject of this report.

SOILSTRUCT-ALPHA (Ebeling, Duncan, and Clough 1990) is a special-purpose finite element program for two-dimensional (2-D) plane strain analysis of soil-structure interaction problems. SOILSTRUCT calculates displacements and stresses resulting from incremental construction, backfilling, excavation, dewatering, rising water table, and/or load application. Nonlinear, stress-path-dependent, stress-strain behavior of the backfill was approximated in the finite element analysis using the tangent modulus method. In the tangent modulus method, new values of tangent moduli are assigned to each soil element at each increment of loading (i.e., dewatering, lock construction, and backfilling) or unloading (i.e., excavation, rising water table). The modulus values assigned to each element are adjusted in accordance with their stresses to simulate nonlinear behavior.

Three types of finite elements are used to represent the behavior of different materials:

- a. *Two-dimensional continua elements.* A 2-D, subparametric, quadrilateral element (QM5) is used to represent the soil and most structural materials. Structural supports, such as the struts or tieback components of an excavation support system, are typically modeled as a spring support using bar elements. However, 2-D elements have been used to model these supports.
- b. *Interface elements.* SOILSTRUCT-ALPHA has the ability to model the interface region between the soil backfill and the structure using interface elements. This important feature allows for the movement of the softer continua elements used in modeling the backfill relative to the movement of the stiffer continua elements used in modeling the structure. This element is defined by four nodes, with each of the two pairs of nodes having the same coordinates; thus this element has no thickness.
- c. *One-dimensional bar elements.* One-dimensional, two-node, bar or spring elements are used to model the behavior of a variety of structural systems. This includes the modeling of structural supports such as braces or tiebacks or the modeling of reinforcement placed within a soil backfill.

SOILSTRUCT was expanded during the U.S. Army Corps of Engineers' first Repair, Evaluation, Maintenance, and Rehabilitation Research Program to model the loss of contact between the base of a wall (a lock in this case) and its rock foundation using a procedure called the ALPHA method (Ebeling, Duncan, and Clough 1990; Ebeling et al. 1992). The ALPHA method was extended to soil elements by Regalado, Duncan, and Clough (1992) to reduce numerical inaccuracies in soil elements that are at or near failure.

The continua elements used to model the soil and the soil-to-structure interface elements that may have failed in shear at one stage of loading have the ability to recover their shear stiffness and shearing resistance as a result of an increase in confining pressures at some later stage of loading in this version of SOILSTRUCT-ALPHA. Several other improvements have been made to the material models, including the new Gomez, Filz, and Ebeling (2000a, 2000b) extended load/unload/reload hyperbolic model for interfaces, and to the numerical procedures implemented within SOILSTRUCT-ALPHA based on experience gained at the U.S. Army Engineer Research and Development Center in conducting soil-structure interaction analyses of different types of Corps structures.

The SOILSTRUCT-ALPHA for Personal Computers analysis package comprises three separate stages: the Visual Modeler, the SOILSTRUCT-ALPHA Finite Element Analysis Program, and the Visual Postprocessor. Figure 1-1 illustrates the relationships among the separate stages of the SOILSTRUCT-ALPHA analysis package.

1.1.1 Visual Modeler

The Visual Modeler is the preprocessing stage for SOILSTRUCT-ALPHA for Personal Computers. The Visual Modeler itself comprises two programs, the Mesh Generator (*MeshGen*) and the Load Generator (*LoadGen*). The Mesh Generator is a computer-aided design program for user-controlled, expedient generation of 2-D finite element meshes. The Mesh Generator is a line-based system that generates three- and four-node elements automatically with nodes computed from the intersection of line segments. To expedite the generation of mesh lines, the user defines the boundaries and material areas of the model with precise coordinates, and then meshes the resulting boundaries with tools that allow connecting the boundary lines with an arbitrary number of lines spaced with a user control based on Beziér-spline methods. The Mesh Generator stores its data in a line-based file with the extension *.lin*, and generates an element file, designated by the extension *.msh*, which is used by the Load Generator. The Mesh Generator was separated from the Load Generator because the Mesh Generator was a more general program and could be used for other 2-D finite element modeling work, and because enhancements could be added more easily, without fear of impacting the load addition methods.

The Visual Modeler's Load Generator is, by nature, specific to the SOILSTRUCT-ALPHA for Personal Computers analysis package. The Load Generator is designed to allow the user to quickly and easily select nodes or elements that will be loaded with any of SOILSTRUCT-ALPHA's ten load types, and to accurately place bar elements. Plotting the elements at an 80 percent scale enables the selection of nodal points relative to elements. Selection is accomplished using Region Assignment, where the user draws a polygon that surrounds the element centers or nodal points, and then selects the attributes assigned to that region. Load cases are viewed with their appropriate loads separately, and the load cases can be added or removed quickly and easily. The Load Generator reads in the *.msh* file exported from the Mesh Generator and stores the loads in an *.lds* file. The program also generates a *.dat* file that is the input for the SOILSTRUCT-ALPHA Finite Element Analysis Program.

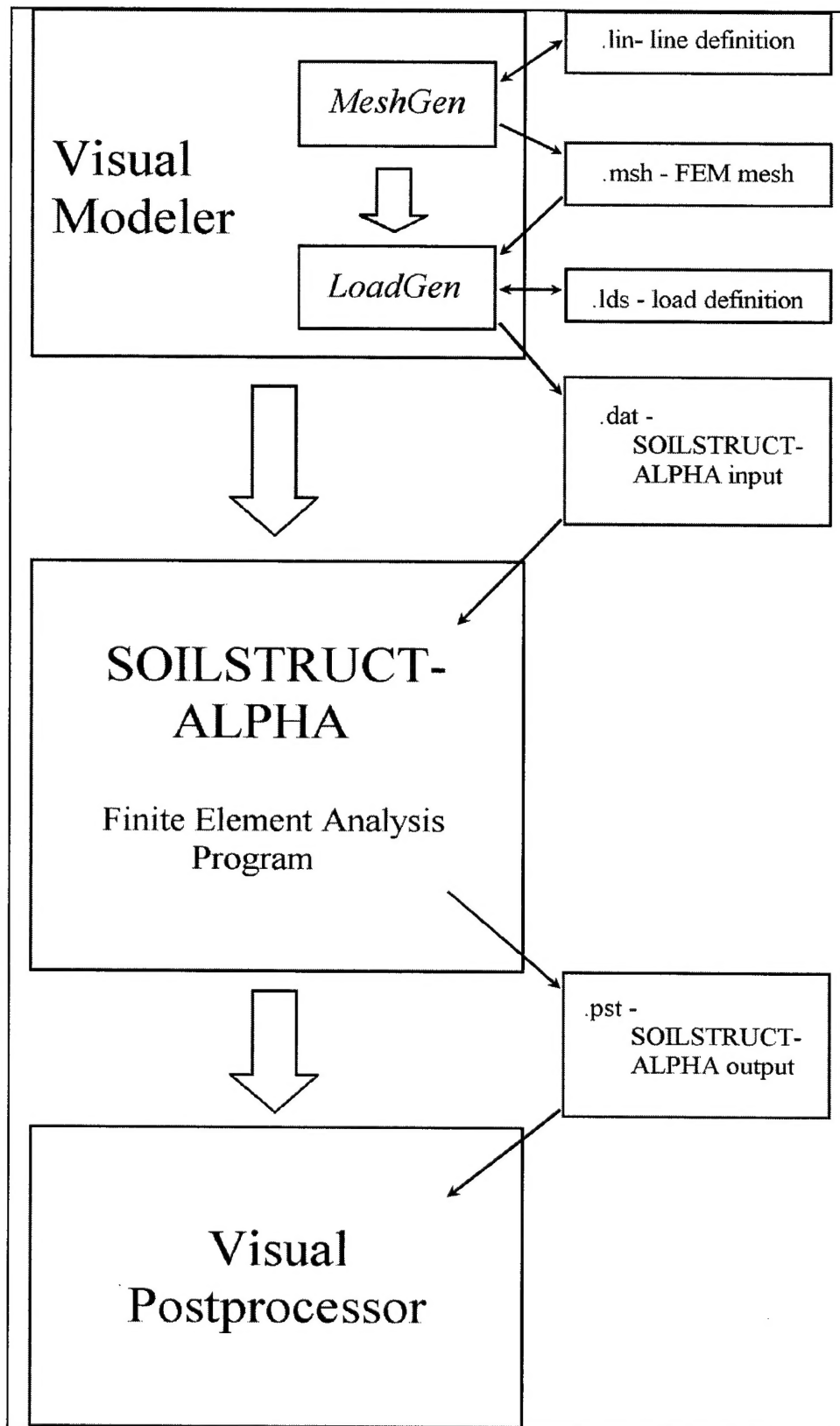


Figure 1-1. Programs for SOILSTRUCT-ALPHA and their associated file extensions

1.1.2 SOILSTRUCT-ALPHA Finite Element Program

The SOILSTRUCT-ALPHA Finite Element Analysis Program is a stand-alone executable version. It requires only a data input file with the extension **.dat**, which is output from the Visual Modeler's load generation program. It also outputs a **.pst** file, which will be used by the Visual Postprocessor.

1.1.3 Visual Postprocessor

The Visual Postprocessor is used to view the results stored in a **.pst** file (i.e., element stresses and nodal point displacements) from the nonlinear finite element analysis of the soil-structure interaction problem modeled using SOILSTRUCT-ALPHA for Personal Computers. This visual postprocessor allows the user to interpret computed results visually.

1.2 Modeling Guidelines

A Visual Modeler user may find the following guidance helpful:

- a. SOILSTRUCT-ALPHA assumes a right-hand coordinate system. It is advisable that the users prepare a sketch of their structure in this coordinate system prior to execution of the Visual Modeler.
- b. In a SOILSTRUCT-ALPHA construction sequence analysis using the backfill placement method of analysis, it is advisable that the user follow the actual geometry and construction sequence as closely as possible when assembling the finite element model and in the subsequent incremental analyses.
- c. In the backfill placement analysis, the backfill is included in the finite element mesh, and the forces acting on the wall are developed through a backfill operation simulation.
- d. In a backfill placement analysis, the backfill operation is simulated by placement of the backfill in a series of lifts. For each lift, the soil placed generates a set of forces that are applied to the wall and previously placed backfill system. With an equivalent linear stress-strain soil model used, the moduli within the existing soil backfill are adjusted to be consistent with the current level of loading. In a backfill placement analysis, two iterations are typically specified for each lift; the ALPHA method is not implemented until the second computation is completed per lift.
- e. Interface elements are placed in the following situations:
 - (1) Between the backfill and the structure to allow for relative movement between the soil and the structure.
 - (2) Between the backfill and the rock foundation (when present) to allow for relative movement between the soil and rock.

- (3) Between the structure and its foundation, be it a soil or rock foundation, to allow for relative movement between the structure and its foundation.
- f. The ALPHA method was developed by Ebeling, Duncan, and Clough (1990) using data computed within structure-to-rock interface elements in a numerical algorithm that allows for base separation to occur between the base of a massive concrete lock wall and its rock foundation. Special requirements are placed on these particular interface elements:
- (1) The base of the lock wall-to-rock foundation interface must be horizontal.
 - (2) The interface elements are numbered consecutively starting with the interface element at the toe of the wall and ending with the interface element at the heel of the wall.

Further details on the ALPHA method are given on pages 64 through 70 in Ebeling et al. (1992).

- g. Cook, Malkus, and Plesha (1989, page 578) note that a 2-D continua element performs best if its shape is compact and regular. They also observe that an element tends to stiffen and lose accuracy as its aspect ratio increases. A typical recommendation is that the aspect ratio for 2-D continua elements be restricted to 5 to 1 for meshes analyzed using a linear elastic finite element method of analysis to compute accurate results. This is reasonable guidance for linear elastic elements that are used in SOILSTRUCT-ALPHA to model structures made of concrete or steel and rock foundations. However, based on experience by the second author of this report, this general guidance is too generous for soil regions of interest in SOILSTRUCT-ALPHA meshes. For example, in an incremental, equivalent linear method of analysis, it is recommended that the aspect ratio of soil elements be nearly 1 to 1 within critical regions such as at the soil-to-structure interface region along the back of the retaining structure. Within other soil regions of less importance, the aspect ratio can be increased. Every attempt is made by the second author of this report to limit the aspect ratio to 3 to 1 for soil elements.
- h. Skewed and triangular 2-D continua elements are acceptable for linear elastic elements used to model a concrete or steel structure or rock foundations. (A skewed element is defined as the corner angle between adjacent element faces that is not equal to 90 degrees. Note that at every corner of both square rectangular elements, the corner angle is exactly 90 degrees.) Cook, Malkus, and Plesha (1989, page 578) observe that an element tends to stiffen and lose accuracy as the corner angles become markedly different from one another. In general, the more skewed a 2-D continua element is, the more likely its stiffness will be numerically (that is, artificially) enhanced. This is an undesirable effect. Again, an element performs best if its shape is compact and regular. This general advice holds true for meshes analyzed using SOILSTRUCT-ALPHA.

- i. For an incremental, equivalent linear method of analysis, it is recommended that the use of triangular elements be minimized in the mesh, especially within soil regions of interest. Triangles should be avoided, if possible, at the soil-to-structure interface region along the back of the retaining structure and other critical regions.
- j. Clough and Duncan (1969) showed that the accuracy in computed results for a backfill placement analysis improves as the number of soil lifts used is increased. In a backfill placement analysis, eight soil lifts are generally accepted as the minimum number of lifts that should be used to maintain sufficient accuracy in computed results. Clough and Duncan also showed (in their Figure 25 for "average stresses") that the magnitude of the error also depends on the magnitude of the hyperbolic soil parameter exponent n , with larger errors associated with larger values of n . In their parametric evaluation, the values for n were between 0 and 1.1. These error data were generated using the results from SOILSTRUCT analyses and closed-form solutions for deflections of a one-dimensional (1-D) representation of incremental fill placement.
- k. Clough and Duncan (1969) also showed that the accuracy in computed results for an excavation analysis improves as the number of soil layers used is increased. They developed a relationship, given in their Figure 24 (for "average stresses"), between the error in simulation of small layer excavation as a function of the number of layers used in the analysis. The magnitude of the error in excavation simulation also depends on the magnitude of the hyperbolic soil parameter exponent n , with larger errors associated with larger values for n . Their Figure 24 was generated using the results from SOILSTRUCT analyses and closed-form solutions for deflections of a 1-D representation of incremental excavation. No general guidance has evolved over the years as to the number of incremental excavation layers that should be used to maintain sufficient accuracy in computed results. However, the second author of this report attempts to use a minimum of 10 to 15 layers in an excavation analysis, depending upon the structure being analyzed.

1.3 Hyperbolic Soil Modeling Parameter Guidelines

SOILSTRUCT-ALPHA uses a hyperbolic representation for nonlinear stress-strain relationships for soil, as described by Duncan and Chang (1970). The use of this mathematical relationship to represent the primary loading curve of soil allows for the development of a mathematical expression for the tangent modulus. Recall that the tangent modulus method combined with the incremental construction method of analysis accounts for the nonlinear stress-strain behavior of soil. The value of the tangent modulus is a function of (a) the stresses computed within each element, (b) the Mohr-Coulomb soil strength parameters, and (c) a group of hyperbolic parameters that are obtained from triaxial soil test data. The stresses computed within each soil element are characterized in terms of the confining pressure and the level of shearing applied to the element of soil. The

confining pressure is expressed in terms of the minor principal stress in the hyperbolic model. The level of shearing applied to the element of soil is expressed in terms of the principal stress difference. The values for confining pressure and the level of shearing applied to a soil element will change during each iteration and in every load step during the course of the analysis. Consequently, the modulus of each soil element is adjusted during each iteration of every load step during the course of the analyses in accordance with the stresses in the element.

A Visual Modeler user may find the following guidance helpful:

- a. The hyperbolic representation for nonlinear stress-strain relationships for soil requires that shear test data be obtained from triaxial tests on high-quality undisturbed soil specimens recovered from each soil stratum in the field. Appendix A in Gomez, Filz, and Ebeling (2000b) outlines the steps involved in the reduction of these data to obtain the Mohr-Coulomb soil strength parameters and the group of hyperbolic parameters from these triaxial soil test data. The determination of the Mohr-Coulomb and hyperbolic model parameters is shown for isotropically consolidated triaxial tests on dense Blacksburg Sand sheared in a drained mode (i.e., isotropically consolidated drained (ICD) tests).
- b. Over the years, a large number of triaxial shear tests for different soils have been performed in which hyperbolic stress-strain model parameters have been extracted. A number of these tabulated data summaries have been collected in Appendix A of Gomez, Filz, and Ebeling (2000b). The tabulated hyperbolic parameter values data contained in this appendix are not intended to replace site-specific testing on high-quality soil specimens, but are for use as interim guidance prior to availability of laboratory test results. These data can be used as an aid for selecting hyperbolic parameter values for analysis when test data are not yet available. It has been observed that the hyperbolic parameters correlate according to soil type and with standard engineering indices.

1.4 Interface Modeling Parameters Guidelines

SOILSTRUCT-ALPHA contains the Gomez, Filz, and Ebeling (2000a, 2000b) extended load/unload/reload hyperbolic model for interfaces. The model is based on the Clough and Duncan (1969) hyperbolic representation for nonlinear shear stress-shear deformation relationship for primary loading, which was extended to model a variety of stress paths. The use of this mathematical relationship to represent the primary loading curve of soil-to-structure interfaces allows for the development of a mathematical expression for the tangent interface stiffness. Recall that the tangent stiffness method combined with the incremental construction method of analysis accounts for the nonlinear shear stress-shear deformation behavior of soil-to-structure interfaces. The value of the tangent interface stiffness is a function of (a) the normal and shear stresses computed within each interface element, (b) the Mohr-Coulomb soil-to-structure interface strength parameters, and (c) a group of hyperbolic parameters that are obtained from soil-to-structure interface test data. The stresses computed within each

interface element are characterized in terms of the normal pressure and the level of shearing applied to the interface element. The values for normal pressure and the level of shearing applied to an interface element will change during each iteration and in every load step during the course of the analysis. Consequently, the stiffness of each interface element is adjusted during each iteration of every load step during the course of the analyses in accordance with the stresses in the interface element. Note that the Gomez, Filz, and Ebeling extended load/unload/reload interface model does not require any material property values in addition to those considered in the Clough and Duncan (1969) hyperbolic interface model.

A Visual Modeler user may find the following guidance helpful:

- a. The hyperbolic representation for nonlinear stress-strain relationships for soil requires that shear test data be obtained from structure-to-soil interface tests. Appendix B in Gomez, Filz, and Ebeling (2000b) outlines the steps involved in the reduction of these data to obtain the Mohr-Coulomb concrete-to-soil interface strength parameters and the group of hyperbolic parameters from these soil-to-structure interface test data. The determination of the Mohr-Coulomb and hyperbolic interface model parameters are shown for concrete-to-soil interface tests on dense Blacksburg Sand. The data given in this appendix are from interface tests performed in the Large Displacement Shear Box (LDSB) at Virginia Polytechnic Institute and State University, Blacksburg. The LDSB soil-to-concrete interfaces can be as large as 711 by 406 mm (28 by 16 in.) allowing a maximum interface displacement as large as 305 mm (12 in.) (Shallenberger and Filz 1996). Gomez, Filz, and Ebeling (2000a) observe that this size direct shear box (DSB) overcomes two of the limitations of traditional DSB: The maximum relative displacement that can be attained in a conventional DSB is limited; hence, the determination of the interface residual strength becomes difficult. In addition, end effects, induced by the presence of the rigid walls of the soil container, may induce errors in the results.
- b. Over the years, a limited number of experimental studies have been conducted to obtain data on the behavior of soil-to-structure interfaces for different soils: Potyondy (1961), Clough and Duncan (1969), Peterson et al. (1976). Their tests were performed using a slightly modified, standard size DSB in which a concrete specimen occupied one of the halves of the shear box. Potyondy (1961) provides Mohr-Coulomb interface shear strength parameters for a variety of structural materials and soils. Clough and Duncan (1969) and Peterson et al. (1976) provide Mohr-Coulomb interface shear strength parameters for the interface between concrete and different types of soils as well as the corresponding hyperbolic interface model parameters for primary loading. Gomez, Filz, and Ebeling (2000a, 2000b) report on LDSB interface tests between concrete and a variety of granular soils. Table 3-8 in Gomez, Filz, and Ebeling (2000b) contains a summary of hyperbolic parameter values for these interface tests. The citations listed in this paragraph for tabulated hyperbolic parameter values are not intended to replace site-specific testing, but are for use as interim guidance prior to availability of laboratory test results. These data can be used as an aid for selecting hyperbolic parameter values for analysis when

test data are not yet available. It has been observed that the hyperbolic interface parameters correlate according to soil type with standard engineering indices (e.g., Peterson, et al. 1976), and with hyperbolic stress-strain soil parameters (Table 3-9 in Gomez, Filz, and Ebeling 2000b).

1.5 Example Structure

An example of a complete soil-structure interaction analysis of a lock wall retaining earth using SOILSTRUCT-ALPHA is given in Ebeling and Wahl (1997). Figure 1-2 shows a portion of the cross section of the new McAlpine roller-compacted concrete lock wall that was analyzed using SOILSTRUCT-ALPHA. This lock structure and its backfill will serve as an example for the step-by-step assembly of a complete SOILSTRUCT-ALPHA input data file using the Visual Modeler. The finite element mesh data will be assembled using *MeshGen*, and the load cases to be applied to the structure are assembled using *LoadGen*. Note that due to figure size limitations for producing this report, the rock foundation that was included in the original analysis is excluded in this demonstration.

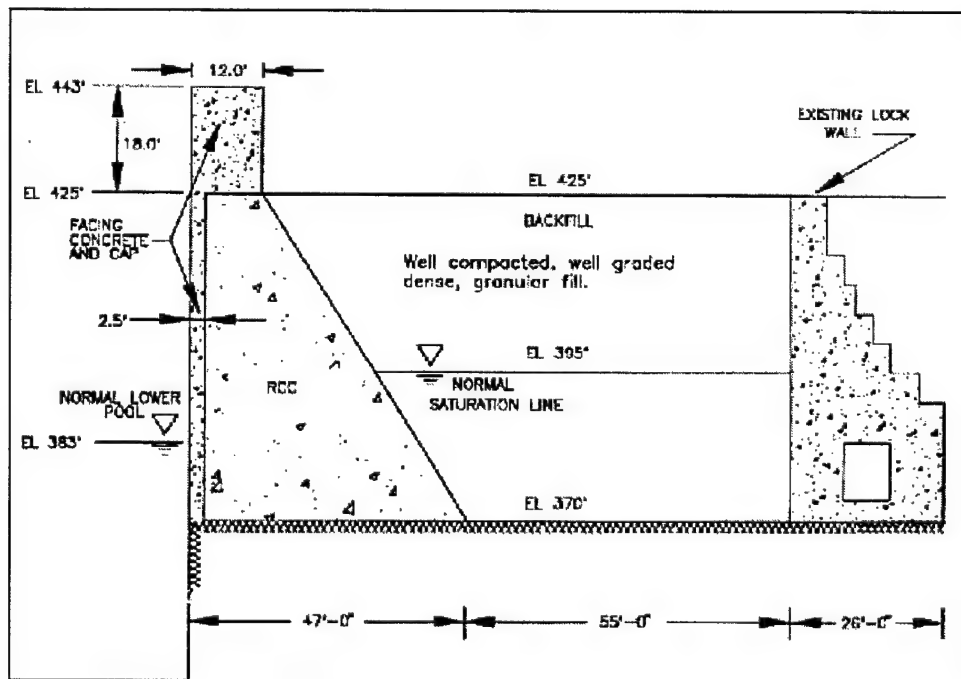


Figure 1-2. McAlpine Lock and Dam analysis (Ebeling and Wahl 1997)

1.6 Nomenclature

For the purposes of this report, the following nomenclature will be used:

Program names will always be italicized.

Definitions: Definitions will be surrounded by a double-lined bounding box.

Filename extensions will be preceded by a dot and use a bold typeface (i.e., **.pst**).

Names of tools will be in bold.

2 *MeshGen*, the Mesh Generator

As stated in Chapter 1, the Mesh Generator is a computer-aided design program for user-controlled, expedient generation of 2-D finite element meshes. This program is a line-based system that generates three- and four-node elements automatically with nodes computed from the intersection of line segments. To expedite the generation of mesh lines, the user defines the boundaries and material areas of the model with precise coordinates, and then meshes the resulting boundaries with tools that allow connecting the boundary lines with an arbitrary number of lines spaced with a user control based on Beziér-spline methods. The Mesh Generator stores its data in a line-based file with the extension **.lin**, and generates an element file, designated by the extension **.msh**, which is used by the Load Generator. The Mesh Generator was separated from the Load Generator because the Mesh Generator was a more general program and could be used for other 2-D finite element work, and because enhancements could be added more easily without fear of impacting the load addition methods.

Zooming: Zooming can be performed at any time by click-dragging a zoom region with the right mouse button. To return the zoom to view the full extent of the model, double-click with the right mouse button. Zooming comes in handy when trying to be precise with that assignment region, and especially for finding that closing handle.

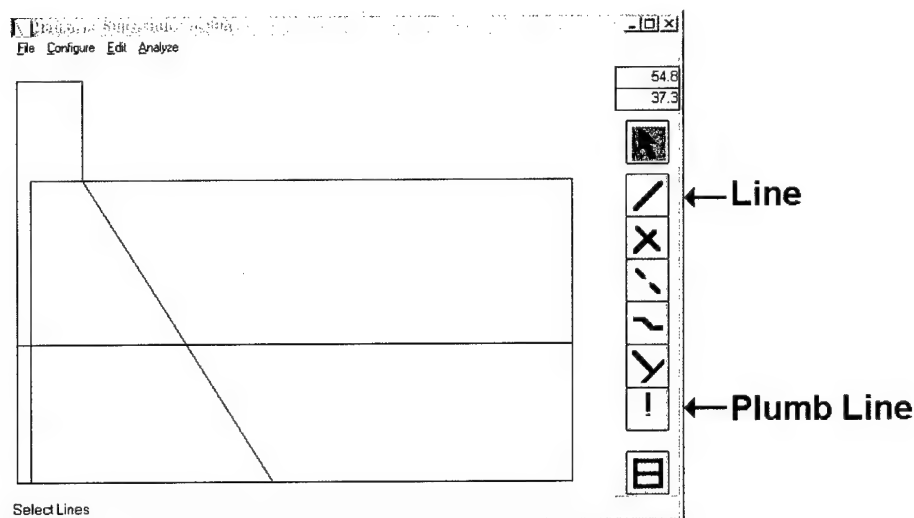
2.1 Step 1: Initialize the Editor for the Model

In this step the user needs to perform the following tasks to set standard units and extents for the newly created model:

- Set the units for the current problem, either SI or non-SI.
- Set the drawing limits for the current problem.
- Specify the snap tolerance of the units.

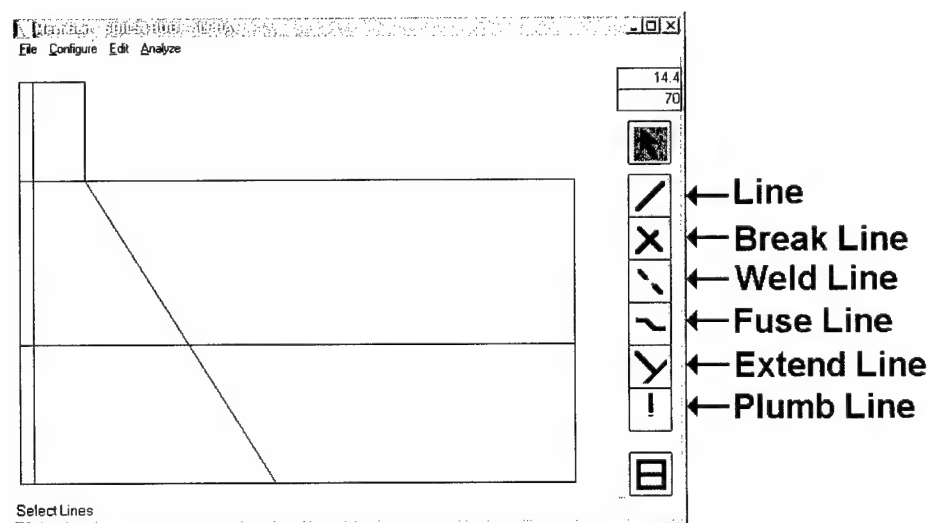
2.2 Step 2: Define the Boundaries of the Model

Using the **Line** and **Plumb Line** tools of the *MeshGen* editor, add the boundary lines for the model. These boundary lines will separate different material areas and the complete boundary of the model.

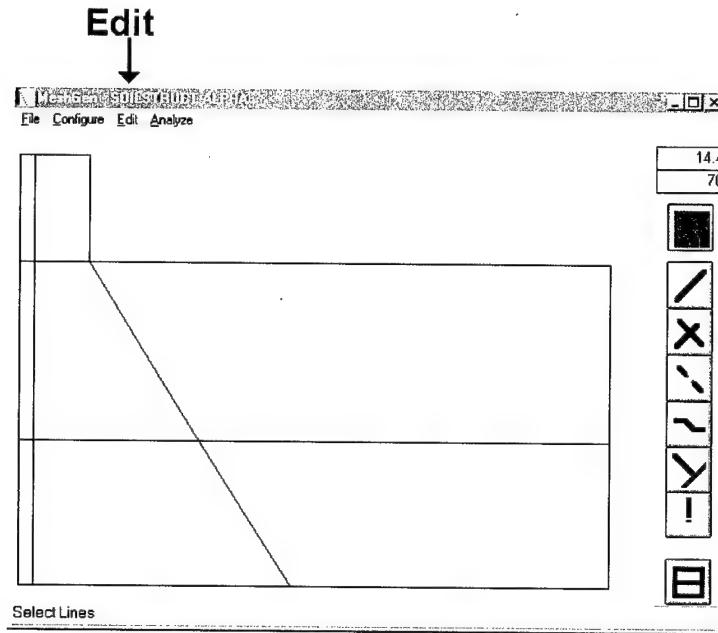


2.3 Step 3: Break the Material Boundaries into Four- or Three-sided Regions

After the boundary definition stage, the model has many different material areas, some of which might be complex. It is best to take these regions and break them into manageable parts. Use the **Line**, **Break Line**, **Weld Line**, **Fuse Line**, **Extend Line**, and **Plumb Line** tools to simplify the regions to help with the meshing tools.

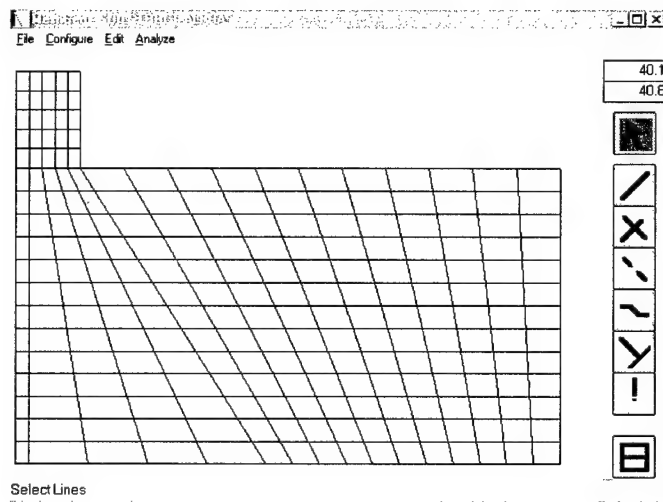
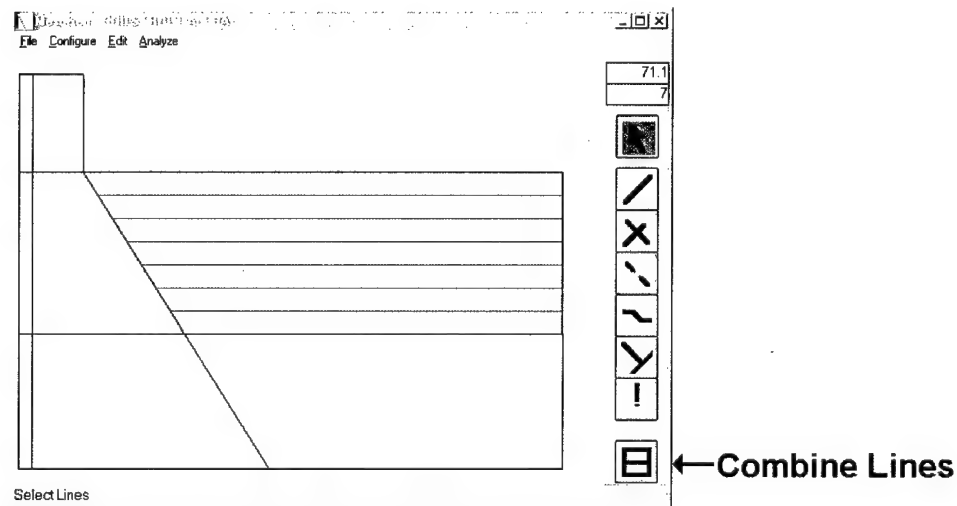


Once everything is in place, use the menu item **Split All Lines** from the **Edit** menu to make line segments that can be conveniently used in Step 4 (Mesh the Regions). This command separates continuous lines into line segments at all the intersection points where lines cross one another.



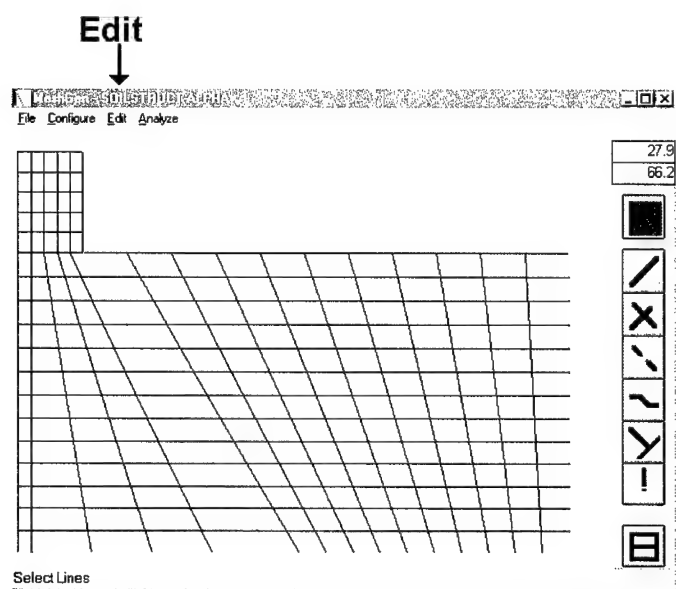
2.4 Step 4: Mesh the Regions

Using the **Combine Lines** tool, select two lines to mesh between. Assign the number of lines and the method (linear or nonlinear) used to connect the selected lines. Continue until the entire region is meshed.



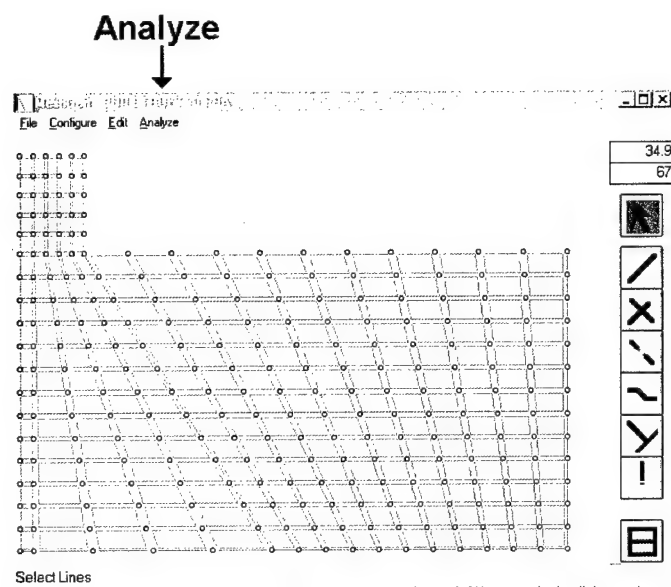
2.5 Step 5: Assign Interface Lines

Certain lines can be marked to become interface elements when the mesh is generated. These lines can be marked by selecting them (using the left mouse button and dragging a region that overlaps or surrounds the line) and by choosing **Set Selected Lines as Interface Elements** from the **Edit** menu. There is also a method to reset a line.



2.6 Step 6: Generate Mesh (which will be input into the Load Generator)

From the **Analyze** menu of *MeshGen*, choose **Generate Mesh**. The mesh generation automatically builds nodes and elements from the line drawings of *MeshGen*.



2.7 Step 7: Optimize Mesh (optional)

Perhaps the project takes a long time to run under SOILSTRUCT-ALPHA, or even does not run because of memory constraints. Sometimes the best solution is to renumber the nodes in the mesh. An efficient node numbering optimizer can reduce the size of the matrices involved in finding a solution. Because it is faster to operate on smaller matrices, the newly renumbered mesh is more time and space efficient. However, the new node configuration is usually nonintuitive, forcing the user to search for the node he or she would like information on. For this reason, there are three ways to number nodes for bandwidth minimization in *MeshGen*.

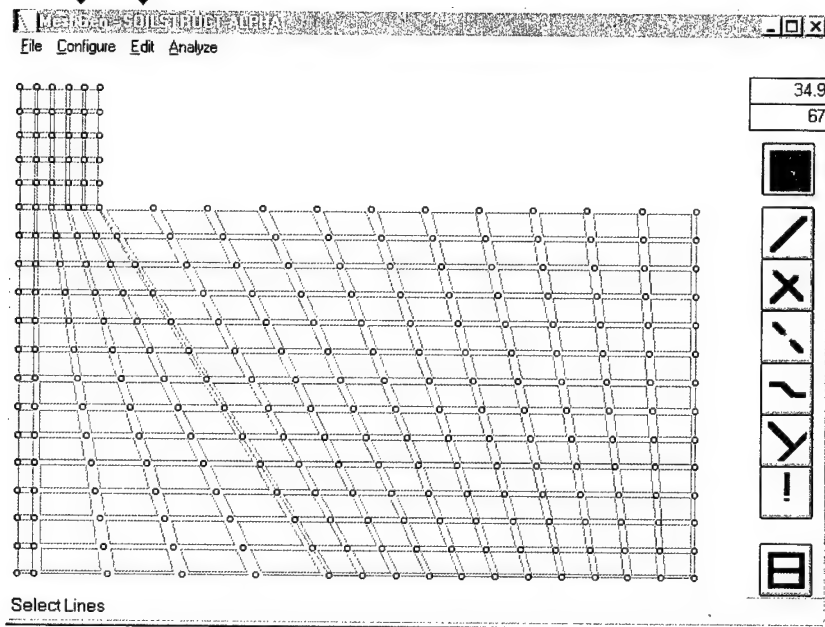
Under the **Analyze** menu of the *MeshGen* program, there is an option labeled **Advancing Wavefront Optimization**. Of the renumbering methods, this option provides the most efficient and least intuitive renumbering for nodes. This renumbering scheme takes connectivity into account when it tries to optimize the mesh.

The other two options deal only with sorting the nodes, whether in the X- or Y-direction first, and in the complementary direction second. If a mesh is taller than it is wide, it is usually a good idea to sort in the X-direction first. If the model is wider than it is tall, then it will generally be better to sort in the Y-direction first.

2.8 Step 8: Verify Mesh – Node Numbers and Element Numbers

From the **Edit** menu, choose **Plot Mesh** to see the newly generated mesh. Selecting **Element Numbers** or **Node Numbers** plots the element and node numbers, respectively. If the elements and nodes are not in the order expected, go to the **Configure** menu item **Sort Mode** to look at different options. If a different option is selected, the mesh will have to be generated again. To aid in the process, using the right mouse button to drag an area will cause the screen to zoom on that region. Double-clicking the mouse will restore the full view. (This option is available throughout the program.)

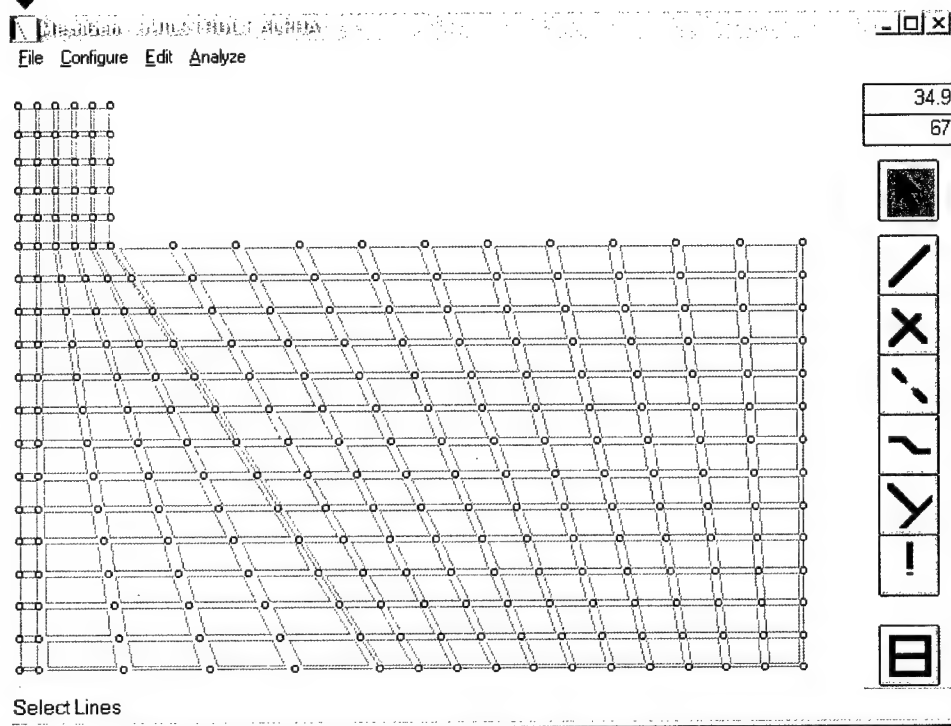
Configure **Edit**



2.9 Step 9: Refine and Reiterate

Save the **.lin** file from the **File** menu and modify the mesh any time. With the editing tools, lines can be deleted, created, or edited over and over again, and the mesh regenerated.

File



3 *LoadGen*, the Load Generator

As stated in Chapter 1, the Visual Modeler's Load Generator is, by nature, specific to the SOILSTRUCT-ALPHA for Personal Computers analysis package. The Load Generator is designed to allow the user to quickly and easily select nodes or elements that will be loaded with any of SOILSTRUCT-ALPHA's ten load types, and to accurately place bar elements. Plotting the elements at an 80 percent scale enables the selection of nodal points relative to elements. Selection is accomplished using Region Assignment, where the user draws a polygon that surrounds the element centers or nodal points, and then selects the attributes assigned to that region. Load cases are viewed with their appropriate loads separately, and the load cases can be added or removed quickly and easily. The Load Generator reads in the **.msh** file exported from the Mesh Generator and stores the loads in an **.lds** file. The program also generates a **.dat** file that is the input for the SOILSTRUCT-ALPHA Finite Element Analysis Program.

Assignment regions: Assignment regions are created by left-clicking with the mouse to create a polygon encompassing either element center(s) or nodal point(s), depending on whatever the load condition affects. The assignment region must be closed to be finished, so the first point selected for the polygon has a handle (drawn as a box) associated with it. Left-clicking with the mouse inside the box closes the polygon and finishes the assignment region. Any number of points can be used to create an assignment region, and there are no restrictions on shape.

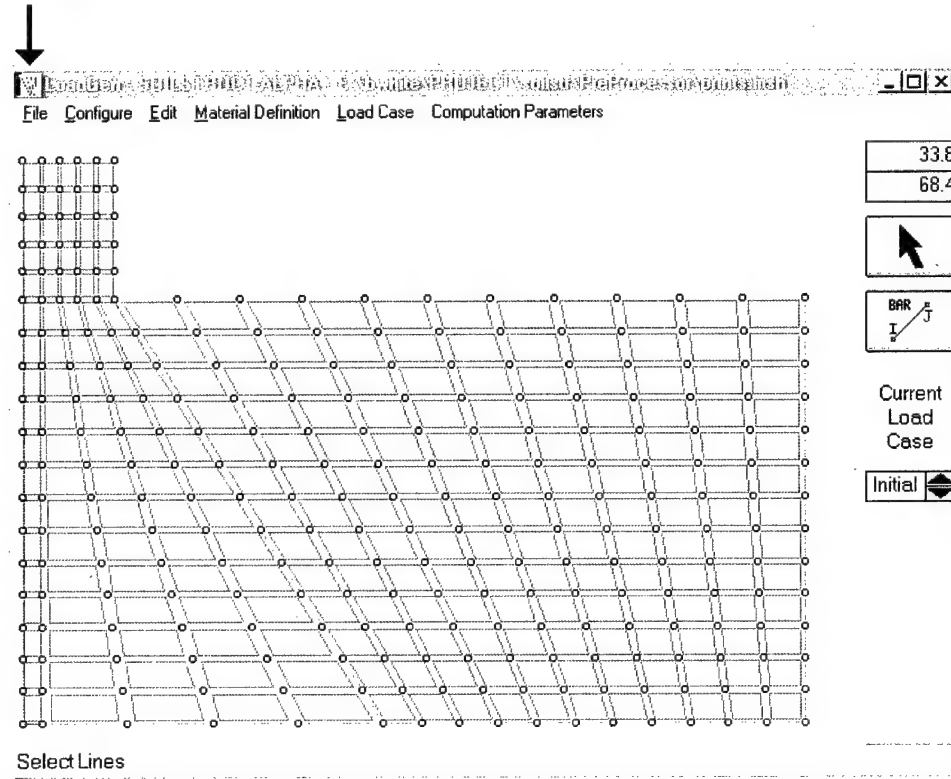
Finishing or canceling an operation: Some operations that can be performed under the Load Assignment preprocessor stage do not have a definite ending point, and some operations might need to be cancelled before they are finished. In those situations, the arrow button on the toolbar to the right of the primary display will display the word "Finished" or "Cancel" and the operation can be concluded by clicking on the button.

Zooming: Zooming can be performed at any time by click-dragging a zoom region with the right mouse button. To return the zoom to view the full extent of the model, double-click with the right mouse button. Zooming comes in handy when trying to be precise with that assignment region, and especially for finding that closing handle.

3.1 Step 1: Load the Mesh created in *MeshGen*

From the **File** menu, choose the **Open (Input Mesh)** option and load the mesh created in the Mesh Generation program.

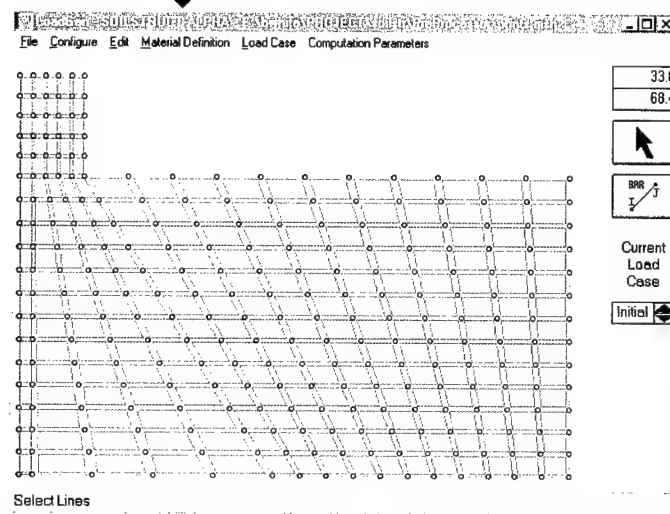
File



3.2 Step 2: Assign Initial Material Properties for 2-D and Interface Elements

From the **Material Definition** menu, choose either the **2D Material** or **Interface Material** submenu. The dialog that appears allows for the entry of the specific properties of each material that will be used in the model. The arrow buttons next to the material number allow the user to cycle through each material and set its properties in turn.

Material Definition

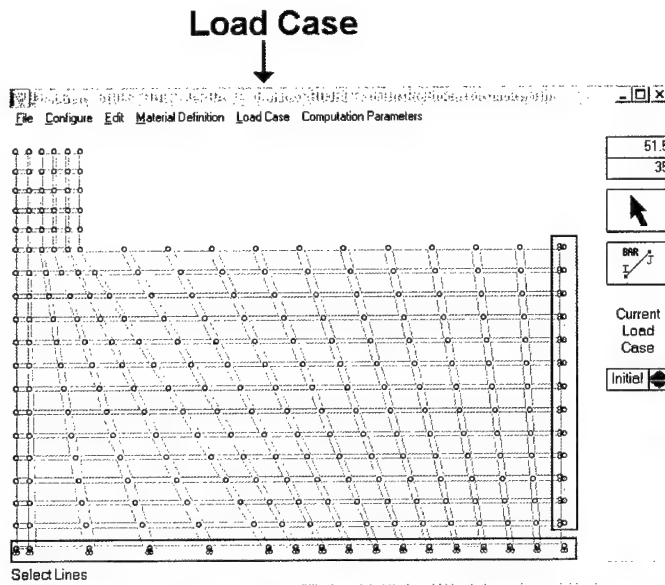


Material Definition	
2D Material Number:	1
Color:	
Material Name:	Air (2D)
Material behavior (IDRAIN):	Patch Element
Poisson's ratio before failure (GUF):	0
Poisson's ratio after failure (GLIF):	0
Total or elongation weight (GAMF):	0
Failure ratio (FR):	0
Coefficient of lateral earth pressure (AO):	0
Friction angle in degrees (PHI):	0
Modulus exponent (n) (XXP):	0
Modulus number (R) (RKJ):	0
Unloaded-reloaded modulus number (R) (RLJ):	0
Undrained shear strength (COJ):	0
Tangent modulus or Young's modulus (E):	0
Tensile strength of interface (TENSJ):	0
Adjacent 2D element material type number (IADJMT):	0
Interface modulus number (RKJ):	0
Interface modulus exponent (XXPJ):	0
<div>Accept</div> <div>Cancel</div>	

Interface Material Definition	
Interface Material Number:	11
Color:	
Interface Material Name:	Air (Interface)
Interface friction angle in degrees (PHI):	0
Min. value for interface shear stiffness (RKS):	0
Interface cohesion (COJ):	0
Failure ratio (FRJ):	0
Tensile strength of interface (TENSJ):	0
Adjacent 2D element material type number (IADJMT):	0
Interface modulus number (RKJ):	0
Interface modulus exponent (XXPJ):	0
<div>Accept</div> <div>Cancel</div>	

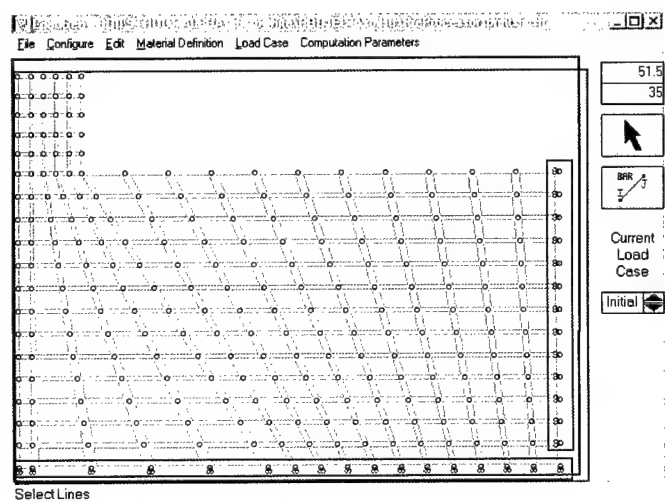
3.3 Step 3: Assign Fixities for the Model

Fixities are assigned to nodal points using assignment regions. From the **Load Case** menu choose the **Assign Fixities** menu item and draw out an assignment region encompassing the nodal points to be fixed. After the assignment region has been selected, a dialog will appear prompting for the type of fixity to apply.



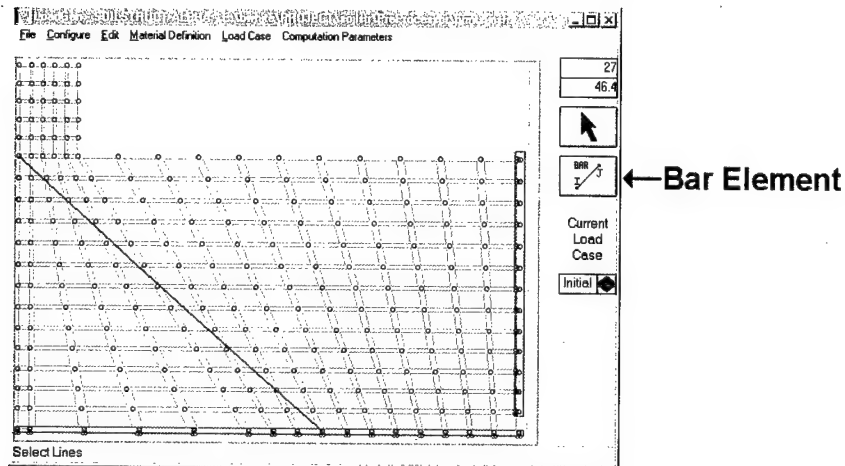
3.4 Step 4: Assign Initial Materials to the Mesh

The initial materials for the mesh's 2-D and interface elements are assigned using assignment regions. From the **Load Case** menu, select either **Assign 2D Element Material** or **Assign Interface Materials**, encompass an assignment region, and select a material from the dialog.



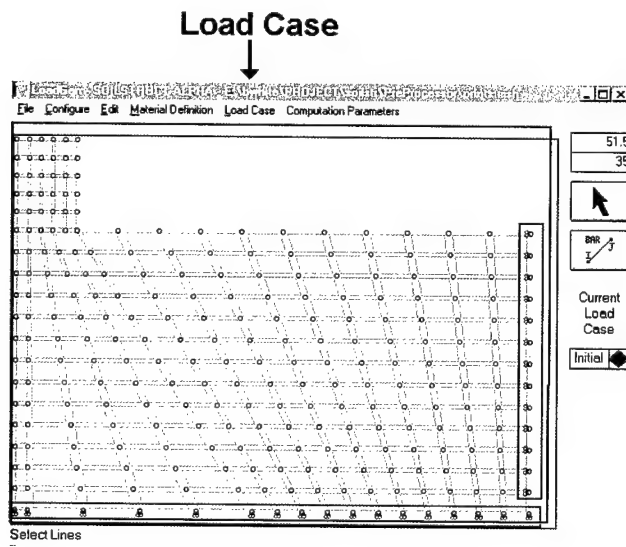
3.5 Step 5: Create Initial Bar Elements (Optional)

Bar elements can be added to the mesh by selecting the **Bar Element** button from the toolbar to the right of the primary viewing screen. Each time the button is pressed, one bar element can be added to the mesh. Since bar elements are connected to a certain node of an element, the node must be selected by click-dragging with the left mouse button over the shrunken element plot's nodal position (i.e., the appropriate corner of the shrunken element). When both end points are selected, a dialog pops up asking for attributes of the bar element.



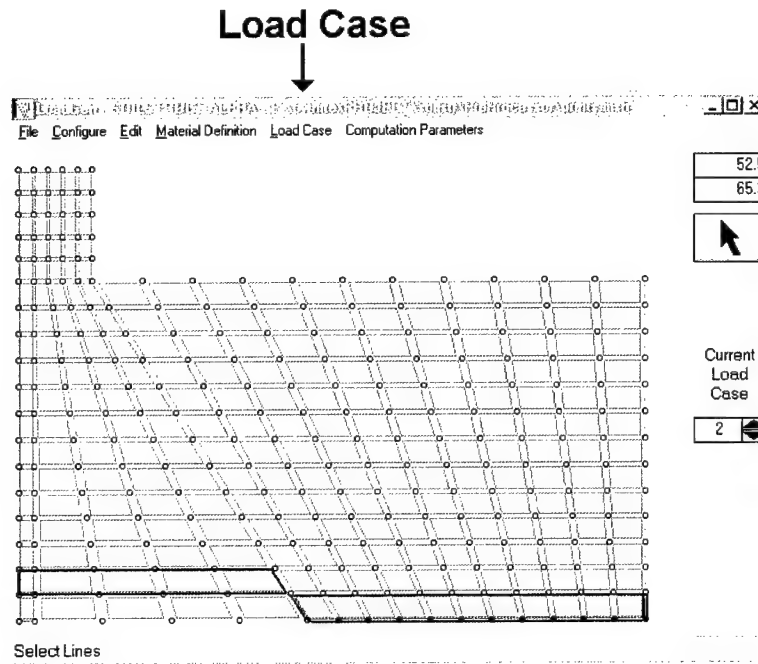
3.6 Step 6: Create Load Cases

Load Cases have to be created on an individual basis. Selecting **Add Load Case** from the **Load Case** menu creates a new load case for the current model. The current load case is accessed/changed through the Current Load Case control on the toolbar to the right of the primary viewing screen. Individual load cases can also be deleted.



3.7 Step 7: Add Regions and Conditions for Each Load Case (Such as Fill Regions, Uplift Conditions, etc.)

By using the **Load Case** menu, up to three types of load conditions can be added to each load case in the model. This is a SOILSTRUCT-ALPHA limitation, but it is strictly enforced in *LoadGen*. When a load type exists in a load case, the load type menu item is flagged for visual confirmation. Most load types can be entered using the region assignment method, with a few exceptions (i.e., bar elements and seepage).



Bar elements can be added in a method similar to creating the initial bar elements. Bar elements can also be deleted by click-dragging with the left mouse button to select the bar element to be deleted.

Seepage is defined by water levels. When **Define Seepage** has been chosen from the **Load Case** menu, a blue line representing water levels is drawn over the model, which has been faded to give a visual cue that the program is in **Define Seepage** mode. Left mouse clicking brings up a dialog box with the current mouse position already entered. If the left mouse has been click-dragged to encompass a nodal point, then that nodal point is entered into the dialog. When the dialog is accepted, the blue line is changed to show how the water level is changed. If the points entered are beyond the extents of the model, then the point is ignored.

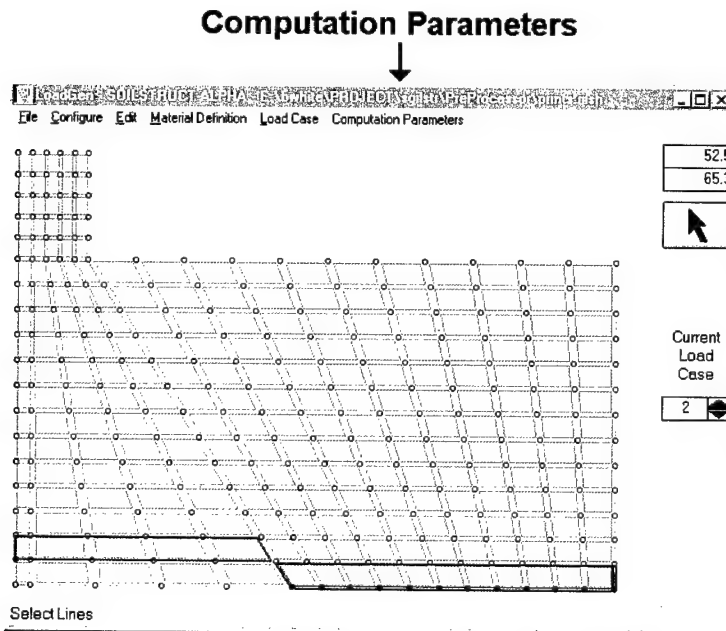
3.8 Step 8: Create Spreadsheet Output (Optional)

SOILSTRUCT-ALPHA has an option to allow the user to output data in a format that spreadsheet programs can read easily. This option allows the user to quickly manipulate and visualize data for specific nodes, elements, and stresses along edges of an element.

To use this option the user needs to select which data he or she desires to output. Selecting **Plot Key** from the **Spreadsheet Output** submenu of the **Computation Parameters** menu causes a dialog to appear. This dialog has a list of options for data output, either stresses for elements, displacements for nodes, normal and shear stresses for interface element edges, or horizontal and vertical stresses for element edges.

After a plot key has been selected for output data, the user needs to select the nodes and elements for output. Selecting **Group Edit** from the **Spreadsheet Output** submenu of the **Computation Parameters** menu starts the node/element selection mode. The user can then create an assignment region about the nodes/elements that he or she wants to use.

If the user has selected stresses for 2-D elements, then any 2-D element handles in the assignment region output those elements. Likewise, if the user has selected displacements for nodes, then all nodes in the assignment region are output. However, if the user selected edge information, then an assignment region must surround an element handle and two nodes along one edge of that element before that information is output. If more than two nodes are surrounded by that region, then each set of nodes for each edge selected is output.



3.9 Step 9: Set Parameters for Analysis (Optional)

SOILSTRUCT-ALPHA gives the user the power to affect how loading calculations are performed. The user has the capability to change parameters for base separation and parameters used to check excessive incremental displacements if the alpha method will be used.

Alpha parameters for base separation are applied by selecting the **Base Separation Parameters** option from the **Computation Parameters** menu. When this option is selected, the user will be prompted for the maximum number of iterations using the alpha method for each load step and for the type of stress increment to be applied using the alpha method. After these parameters have been entered, the user will be required to identify the base interface element group for separation using the alpha method. When the alpha method is invoked, the interface elements forming the base of lock wall-to-rock foundation interface must be horizontal. The LoadGen program will automatically form interface element groups by collecting linear interface element segments. To select a group, a region must be drawn encompassing the group (or groups). Selecting the base group requires the user to select the single horizontal group for separation. After the user has selected the base group, he or she will be prompted to select other groups for which resultant forces and their points of application will be computed. Double-clicking after the resultant groups are selected returns the user to the default editing mode.

Control parameters for checking excessive incremental displacements can be applied by the user by selecting the **Excessive Incremental Displacement Parameters** option from the **Computation Parameters** menu. This option is used when the alpha method is invoked. This menu selection causes a dialog box to appear, prompting the user for information about the excessive incremental parameters. The user is prompted for the type of incremental displacement check to perform, the initial load number for this check, tolerance values for the X- and Y-directions, and whether the program will stop when a tolerance is exceeded.

After the dialog box has been completed, the user will be allowed to select nodal points for which the incremental displacement check is performed. The user creates a region enclosing the nodes to check or double-clicks to choose all of the nodes. If a region has been drawn, all nodal points in that region are checked.

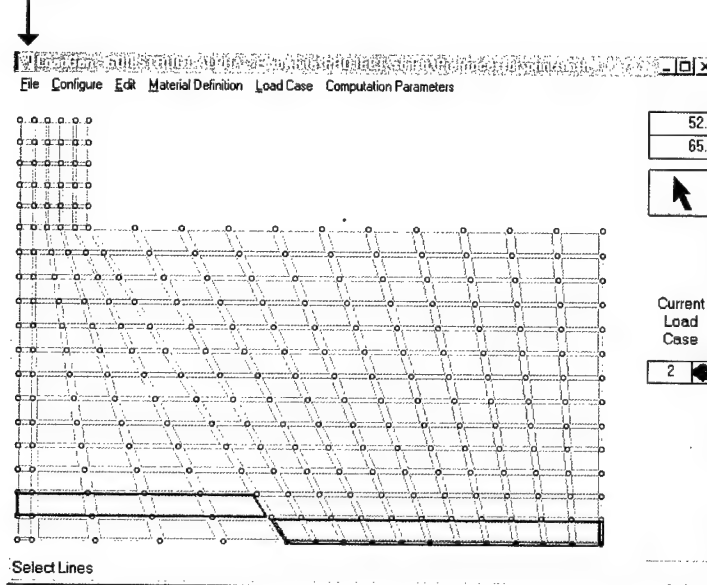
3.10 Step 10: Set Restart Information (Optional)

SOILSTRUCT-ALPHA provides the user with the option of continuing a previous analysis. In order for the user to use this option, he or she must specify that a restart is taking place and specify the analysis results restart file name of the analysis to be restarted. Access the **Input Restart File** submenu from the **File** menu to give the restart information.

3.11 Step 11: Save the Loads into an .lds File

When the load cases have been entered for a specific model, the load cases can be saved in a separate file for easy retrieval and editing. From the **File** menu, select **Save (Loads)**. The load cases will be saved with the same name as the mesh file, but with the extension **.lds**. Bear in mind that if the mesh has changed, the load case file associated with it may be invalid. However, if the mesh has not changed, then its load cases can be reopened and edited easily for refinement of the analysis.

File



3.12 Step 12: Create SOILSTRUCT-ALPHA Input File

This is the final step of the preprocessor stage. When the load cases have been entered for a specific mesh, and the output options selected, a valid SOILSTRUCT-ALPHA input file can be created for processing. Select **Create SOILSTRUCT-ALPHA Input** from the **File** menu to create a **.dat** file for input into the SOILSTRUCT-ALPHA processor.

4 Summary

This report describes how the SOILSTRUCT-ALPHA visual modeler (i.e., a preprocessor) is used, applying it to a suggested work flow of a typical project. The SOILSTRUCT-ALPHA for Personal Computers analysis package comprises three separate stages: the Visual Modeler, the SOILSTRUCT-ALPHA Finite Element Analysis Program, and the Visual Postprocessor. Figure 4-1 illustrates the relationships among the separate stages of the SOILSTRUCT-ALPHA analysis package. The Visual Modeler is intended to expedite the mesh construction and load generation for the typical SOILSTRUCT-ALPHA input file.

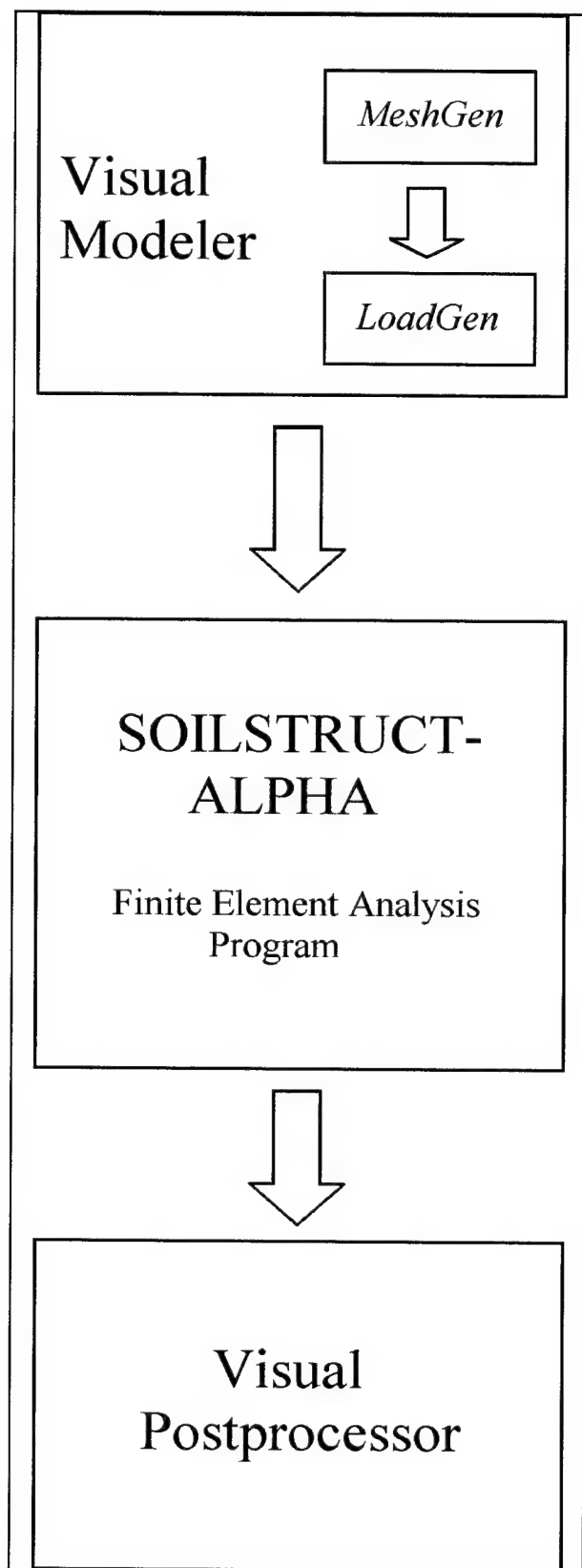


Figure 4-1. SOILSTRUCT-ALPHA Package for Personal Computers

References

- Clough, G. W., and Duncan, J. M. (1969). "Finite element analyses of Port Allen and Old River Locks," Contract Report S-69-6, U.S. Army Engineer Waterways Experiment Station, Vicksburg, MS.
- Cook, R. D., Malkus, D. S., and Plesha, M. E. (1989). *Concepts and applications of finite element analysis*. University of Wisconsin-Madison.
- Duncan, J. M., and Chang, C. Y. (1970). "Nonlinear analysis of stress and strain in soils," *Journal of the Soil Mechanics and Foundations Division*, ASCE, 96(SM5), 1629-1653.
- Duncan, J. M., and Clough, G. W. (1971). "Finite element analyses of Port Allen Lock," *Journal of the Soil Mechanics and Foundations Division*, ASCE, 97(SM8), 1053-1068.
- Ebeling, R. M., and Wahl, R. E. (1997). "Soil-structure-foundation interaction analysis of new roller-compacted concrete, north lock wall at McAlpine Locks," Technical Report ITL-97-5, U.S. Army Engineer Waterways Experiment Station, Vicksburg, MS.
- Ebeling, R. M., Duncan, J. M., and Clough, G. W. (1990). "Methods of evaluating the stability and safety of gravity earth retaining structures founded on rock – phase 2 study," Technical Report ITL-90-7, U.S. Army Engineer Waterways Experiment Station, Vicksburg, MS.
- Ebeling, R. M., Clough, G. W., Duncan, J. M., and Brandon, T. L. (1992). "Methods of evaluating the stability and safety of gravity earth retaining structures founded on rock," Technical Report REMR-CS-29, U.S. Army Engineer Waterways Experiment Station, Vicksburg, MS.
- Gomez, J. E., Filz, G. M., and Ebeling, R. M. (2000a). "Development of an improved numerical model for concrete-to-soil interfaces in soil-structure interaction analyses," Technical Report ITL-99-1, U.S. Army Engineer Research and Development Center, Vicksburg, MS.

- Gomez, J. E., Filz, G. M., and Ebeling, R. M. (2000b). "Extended load/unload/reload hyperbolic model for interfaces: Parameter values and model performance for the contact between concrete and coarse sand," Technical Report ERDC/ITL TR-00-7, U.S. Army Engineer Research and Development Center, Vicksburg, MS.
- Peterson, M. S., Kulhavy, F. H., Nucci, L. R., and Wasil, B. A. (1976). "Stress-deformation behavior of soil-concrete interfaces," Contract Report B-49 to Niagara Mohawk Power Corporation, Syracuse, NY.
- Potyondy, J. G. (1961). "Skin friction between various soils and construction materials," *Géotechnique* 11(4), 339-353.
- Regalado, L. R., Duncan, J. M., and Clough, G. W. (1992). "Finite element analyses of gravity earth retaining structures founded on soil," Contract report to the U.S. Army Engineer Waterways Experiment Station, Vicksburg, MS.
- Shallenberger, W. C., and Filz, G. M. (1996). "Interface strength determination using a large displacement shear box." *Proceedings of the Second International Congress on Environmental Geotechnics*, Osaka, Japan, 5-8 November 1996. M. Kamon, ed., A. A. Balkema, Rotterdam.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing this collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number. **PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.**

1. REPORT DATE (DD-MM-YYYY) December 2001		2. REPORT TYPE Report 1 of a series		3. DATES COVERED (From - To)	
4. TITLE AND SUBTITLE SOILSTRUCT-ALPHA for Personal Computers; Report 1: Visual Modeler				5a. CONTRACT NUMBER	
				5b. GRANT NUMBER	
				5c. PROGRAM ELEMENT NUMBER	
6. AUTHOR(S) Barry C. White, Robert M. Ebeling				5d. PROJECT NUMBER	
				5e. TASK NUMBER	
				5f. WORK UNIT NUMBER 31589	
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Computer Systems Technology, Inc. 3530 Manor Drive, Suite 5 Vicksburg, MS 39180; U.S. Army Engineer Research and Development Center Information Technology Laboratory 3909 Halls Ferry Road Vicksburg, MS 39180-6199				8. PERFORMING ORGANIZATION REPORT NUMBER ERDC/ITL TR-01-2	
9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES) U.S. Army Corps of Engineers Washington, DC 20314-1000				10. SPONSOR/MONITOR'S ACRONYM(S)	
				11. SPONSOR/MONITOR'S REPORT NUMBER(S)	
12. DISTRIBUTION / AVAILABILITY STATEMENT Approved for public release; distribution is unlimited.					
13. SUPPLEMENTARY NOTES					
14. ABSTRACT The SOILSTRUCT-ALPHA for Personal Computers analysis package comprises three separate stages: the Visual Modeler, the SOILSTRUCT-ALPHA Finite Element Analysis Program, and the Visual Postprocessor. This report describes how the SOILSTRUCT-ALPHA Visual Modeler (i.e., a preprocessor) is used, applying it to the suggested work flow of a typical project as an example. This report is intended to serve as a user's guide and not as a reference manual defining each specific command. SOILSTRUCT-ALPHA is a special-purpose finite element program for two-dimensional, plane strain analysis of soil-structure interaction problems. SOILSTRUCT calculates displacements and stresses resulting from <u>incremental</u> construction, backfilling, excavation, dewatering, rising water table, and/or load application. Nonlinear, stress-path-dependent, stress-strain behavior of the backfill was approximated in the finite element analysis using the tangent modulus method. In the tangent modulus method, new values of tangent moduli are assigned to each soil element at each increment of loading (i.e., dewatering, lock construction, and backfilling) or unloading (i.e., excavation, rising water table). The modulus values assigned to each element are adjusted in accordance with their stresses to simulate nonlinear behavior. <div style="text-align: right;">(Continued)</div>					
15. SUBJECT TERMS Computer-aided design Finite element modeler Earth retaining structure Nonlinear finite element analysis Finite element analysis Soil-structure interaction					
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES 45	19a. NAME OF RESPONSIBLE PERSON
a. REPORT UNCLASSIFIED	b. ABSTRACT UNCLASSIFIED	c. THIS PAGE UNCLASSIFIED			19b. TELEPHONE NUMBER (include area code)

14. (Concluded).

The Visual Modeler is designed to facilitate the development of a finite element mesh for SOILSTRUCT-ALPHA in a timely manner. The modeler comprises two parts, a computer-aided design tool for the actual creation and refinement of a mesh, including four-noded two-dimensional continua and interface elements, and a computer-aided design tool for the application of loads to the mesh and the creation of one-dimensional, two-noded bar elements. These tools expedite the creation and refinement of the often extensive input data to SOILSTRUCT-ALPHA.

REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

	Title	Date
Technical Report K-78-1	List of Computer Programs for Computer-Aided Structural Engineering	Feb 1978
Instruction Report 0-79-2	User's Guide: Computer Program with Interactive Graphics for Analysis of Plane Frame Structures (CFRAME)	Mar 1979
Technical Report K-80-1	Survey of Bridge-Oriented Design Software	Jan 1980
Technical Report K-80-2	Evaluation of Computer Programs for the Design/Analysis of Highway and Railway Bridges	Jan 1980
Instruction Report K-80-1	User's Guide: Computer Program for Design/Review of Curvi-linear Conduits/Culverts (CURCON)	Feb 1980
Instruction Report K-80-3	A Three-Dimensional Finite Element Data Edit Program	Mar 1980
Instruction Report K-80-4	A Three-Dimensional Stability Analysis/Design Program (3DSAD) Report 1: General Geometry Module Report 3: General Analysis Module (CGAM) Report 4: Special-Purpose Modules for Dams (CDAMS)	Jun 1980 Jun 1982 Aug 1983
Instruction Report K-80-6	Basic User's Guide: Computer Program for Design and Analysis of Inverted-T Retaining Walls and Floodwalls (TWDA)	Dec 1980
Instruction Report K-80-7	User's Reference Manual: Computer Program for Design and Analysis of Inverted-T Retaining Walls and Floodwalls (TWDA)	Dec 1980
Technical Report K-80-4	Documentation of Finite Element Analyses Report 1: Longview Outlet Works Conduit Report 2: Anchored Wall Monolith, Bay Springs Lock	Dec 1980 Dec 1980
Technical Report K-80-5	Basic Pile Group Behavior	Dec 1980
Instruction Report K-81-2	User's Guide: Computer Program for Design and Analysis of Sheet Pile Walls by Classical Methods (CSHTWAL) Report 1: Computational Processes Report 2: Interactive Graphics Options	Feb 1981 Mar 1981
Instruction Report K-81-3	Validation Report: Computer Program for Design and Analysis of Inverted-T Retaining Walls and Floodwalls (TWDA)	Feb 1981
Instruction Report K-81-4	User's Guide: Computer Program for Design and Analysis of Cast-in-Place Tunnel Linings (NEWTUN)	Mar 1981
Instruction Report K-81-6	User's Guide: Computer Program for Optimum Nonlinear Dynamic Design of Reinforced Concrete Slabs Under Blast Loading (CBARCS)	Mar 1981
Instruction Report K-81-7	User's Guide: Computer Program for Design or Investigation of Orthogonal Culverts (CORTCUL)	Mar 1981
Instruction Report K-81-9	User's Guide: Computer Program for Three-Dimensional Analysis of Building Systems (CTABS80)	Aug 1981
Technical Report K-81-2	Theoretical Basis for CTABS80: A Computer Program for Three-Dimensional Analysis of Building Systems	Sep 1981

(Continued)

REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

	Title	Date
Instruction Report K-82-6	User's Guide: Computer Program for Analysis of Beam-Column Structures with Nonlinear Supports (CBEAMC)	Jun 1982
Instruction Report K-82-7	User's Guide: Computer Program for Bearing Capacity Analysis of Shallow Foundations (CBEAR)	Jun 1982
Instruction Report K-83-1	User's Guide: Computer Program with Interactive Graphics for Analysis of Plane Frame Structures (CFRAME)	Jan 1983
Instruction Report K-83-2	User's Guide: Computer Program for Generation of Engineering Geometry (SKETCH)	Jun 1983
Instruction Report K-83-5	User's Guide: Computer Program to Calculate Shear, Moment, and Thrust (CSMT) from Stress Results of a Two-Dimensional Finite Element Analysis	Jul 1983
Technical Report K-83-1	Basic Pile Group Behavior	Sep 1983
Technical Report K-83-3	Reference Manual: Computer Graphics Program for Generation of Engineering Geometry (SKETCH)	Sep 1983
Technical Report K-83-4	Case Study of Six Major General-Purpose Finite Element Programs	Oct 1983
Instruction Report K-84-2	User's Guide: Computer Program for Optimum Dynamic Design of Nonlinear Metal Plates Under Blast Loading (CSDOOR)	Jan 1984
Instruction Report K-84-7	User's Guide: Computer Program for Determining Induced Stresses and Consolidation Settlements (CSETT)	Aug 1984
Instruction Report K-84-8	Seepage Analysis of Confined Flow Problems by the Method of Fragments (CFRAG)	Sep 1984
Instruction Report K-84-11	User's Guide for Computer Program CGFAG, Concrete General Flexure Analysis with Graphics	Sep 1984
Technical Report K-84-3	Computer-Aided Drafting and Design for Corps Structural Engineers	Oct 1984
Technical Report ATC-86-5	Decision Logic Table Formulation of ACI 318-77, Building Code Requirements for Reinforced Concrete for Automated Constraint Processing, Volumes I and 11	Jun 1986
Technical Report ITL-87-2	A Case Committee Study of Finite Element Analysis of Concrete Flat Slabs	Jan 1987
Instruction Report ITL-87-2 (Revised)	User's Guide for Concrete Strength Investigation and Design (CASTR) in Accordance with ACI 318-89	Mar 1992
Instruction Report ITL-87-1	User's Guide: Computer Program for Two-Dimensional Analysis of U-Frame Structures (CUFRAM)	Apr 1987
Instruction Report ITL-87-2	User's Guide: For Concrete Strength Investigation and Design (CASTR) in Accordance with ACI 318-83	May 1987
Technical Report ITL-87-6	Finite-Element Method Package for Solving Steady-State Seepage Problems	May 1987

(Continued)

REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

	Title	Date
Instruction Report ITL-87-3	User's Guide: A Three-Dimensional Stability Analysis/Design Program (3DSAD) Module	Jun 1987
	Report 1: Revision 1: General Geometry	Jun 1987
	Report 2: General Loads Module	Sep 1989
	Report 6: Free-Body Module	Sep 1989
Instruction Report ITL-87-4	User's Guide: 2-D Frame Analysis Link Program (LINK2D)	Jun 1987
Technical Report ITL-87-4	Finite Element Studies of a Horizontally Framed Miter Gate	Aug 1987
	Report 1: Initial and Refined Finite Element Models (Phases A, B, and C), Volumes I and II	
	Report 2: Simplified Frame Model (Phase D)	
	Report 3: Alternate Configuration Miter Gate Finite Element Studies-Open Section	
	Report 4: Alternate Configuration Miter Gate Finite Element Studies-Closed Sections	
	Report 5: Alternate Configuration Miter Gate Finite Element Studies-Additional Closed Sections	
	Report 6: Elastic Buckling of Girders in Horizontally Framed Miter Gates	
	Report 7: Application and Summary	
Instruction Report GL-87-1	User's Guide: UTEXAS2 Slope-Stability Package; Volume 1, User's Manual	Aug 1987
Instruction Report ITL-87-5	Sliding Stability of Concrete Structures (CSLIDE)	Oct 1987
Instruction Report ITL-87-6	Criteria Specifications for and Validation of a Computer Program for the Design or Investigation of Horizontally Framed Miter Gates (CMITER)	Dec 1987
Technical Report ITL-87-8	Procedure for Static Analysis of Gravity Dams Using the Finite Element Method - Phase Ia	Jan 1988
Instruction Report ITL-88-1	User's Guide: Computer Program for Analysis of Planar Grid Structures (CGRID)	Feb 1988
Technical Report ITL-88-1	Development of Design Formulas for Ribbed Mat Foundations on Expansive Soils	Apr 1988
Technical Report ITL-88-2	User's Guide: Pile Group Graphics Display (CPGG) Postprocessor to CPGA Program	Apr 1988
Instruction Report ITL-88-2	User's Guide for Design and Investigation of Horizontally Framed Miter Gates (CMITER)	Jun 1988
Instruction Report ITL-88-4	User's Guide for Revised Computer Program to Calculate Shear, Moment, and Thrust (CSMT)	Sep 1988
Instruction Report GL-87-1	User's Guide: UTEXAS2 Slope-Stability Package; Volume 11, Theory	Feb 1989
Technical Report ITL-89-3	User's Guide: Pile Group Analysis (CPGA) Computer Group	Jul 1989

(Continued)

REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

	Title	Date
Technical Report ITL-89-4	CBASIN-Structural Design of Saint Anthony Falls Stilling Basins According to Corps of Engineers Criteria for Hydraulic Structures; Computer Program X0098	Aug 1989
Technical Report ITL-89-5	CCHAN-Structural Design of Rectangular Channels According to Corps of Engineers Criteria for Hydraulic Structures; Computer Program X0097	Aug 1989
Technical Report ITL-89-6	The Response-Spectrum Dynamic Analysis of Gravity Dams Using the Finite Element Method; Phase 11	Aug 1989
Contract Report ITL-89-1	State of the Art on Expert Systems Applications in Design, Construction, and Maintenance of Structures	Sep 1989
Instruction Report ITL-90-1	User's Guide: Computer Program for Design and Analysis of Sheet Pile Walls by Classical Methods (CWALSHT)	Feb 1990
Instruction Report ITL-90-2	User's Guide: Pile Group-Concrete Pile Analysis Program (CPGC) Preprocessor to CPGA Program	Jun 1990
Instruction Report ITL-90-3	Investigation and Design of U-Frame Structures Using Program CUFRBC	
	Volume A: Program Criteria and Documentation	May 1990
	Volume B: User's Guide for Basins	May 1990
	Volume C: User's Guide for Channels	May 1990
Instruction Report ITL-90-6	User's Guide: Computer Program for Two-Dimensional Analysis of U-Frame or W-Frame Structures (CWFRAM)	Sep 1990
Technical Report ITL-91-3	Application of Finite Element, Grid Generation, and Scientific Visualization Techniques to 2-D and 3-D Seepage and Groundwater Modeling	Sep 1990
Instruction Report ITL-91-1	User's Guide: Computer Program for Design and Analysis of Sheet-Pile Walls by Classical Methods (CWALSHT) Including Rowe's Moment Reduction	Oct 1991
Technical Report ITL-92-2	Finite Element Modeling of Welded Thick Plates for Bonneville Navigation Lock	May 1992
Technical Report ITL-92-4	Introduction to the Computation of Response Spectrum for Earthquake Loading	Jun 1992
Instruction Report ITL-92-3	Concept Design Example, Computer-Aided Structural Modeling (CASM)	
	Report 1: Scheme A	Jun 1992
	Report 2: Scheme B	Jun 1992
	Report 3: Scheme C	Jun 1992
Instruction Report ITL-92-4	User's Guide: Computer-Aided Structural Modeling (CASM) – Version 3.00	Apr 1992
Instruction Report ITL-92-5	Tutorial Guide: Computer-Aided Structural Modeling (CASM) – Version 3.00	Apr 1992

(Continued)

REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

	Title	Date
Contract Report ITL-92-1	Optimization of Steel Pile Foundations Using Optimality Criteria	Jun 1992
Technical Report ITL-92-7	Refined Stress Analysis of Melvin Price Locks and Dam	Sep 1992
Contract Report ITL-92-2	Knowledge-Based Expert System for Selection and Design of Retaining Structures	Sep 1992
Contract Report ITL-92-3	Evaluation of Thermal and Incremental Construction Effects for Monoliths AL-3 and AL-5 of the Melvin Price Locks and Dam	Sep 1992
Instruction Report GL-87-1	User's Guide: UTEXAS3 Slope-Stability Package; Volume IV, User's Manual	Nov 1992
Technical Report ITL-92-11	The Seismic Design of Waterfront Retaining Structures	Nov 1992
Technical Report ITL-92-12	Computer-Aided, Field-Verified Structural Evaluation	
	Report 1: Development of Computer Modeling Techniques for Miter Lock Gates	Nov 1992
	Report 2: Field Test and Analysis Correlation at John Hollis Bankhead Lock and Dam	Dec 1992
	Report 3: Field Test and Analysis Correlation of a Vertically Framed Miter Gate at Emsworth Lock and Dam	Dec 1993
Instruction Report GL-87-1	Users Guide: UTEXAS3 Slope-Stability Package; Volume III, Example Problems	Dec 1992
Technical Report ITL-93-1	Theoretical Manual for Analysis of Arch Dams	Jul 1993
Technical Report ITL-93-2	Steel Structures for Civil Works, General Considerations for Design and Rehabilitation	Aug 1993
Technical Report ITL-93-3	Soil-Structure Interaction Study of Red River Lock and Dam No. 1 Subjected to Sediment Loading	Sep 1993
Instruction Report ITL-93-3	User's Manual-ADAP, Graphics-Based Dam Analysis Program	Aug 1993
Instruction Report ITL-93-4	Load and Resistance Factor Design for Steel Miter Gates	Oct 1993
Technical Report ITL-94-2	User's Guide for the Incremental Construction, Soil-Structure Interaction Program SOILSTRUCT with Far-Field Boundary Elements	Mar 1994
Instruction Report ITL-94-1	Tutorial Guide: Computer-Aided Structural Modeling (CASM); Version 5.00	Apr 1994
Instruction Report ITL-94-2	User's Guide: Computer-Aided Structural Modeling (CASM); Version 5.00	Apr 1994
Technical Report ITL-94-4	Dynamics of Intake Towers and Other MDOF Structures Under Earthquake Loads: A Computer-Aided Approach	Jul 1994
Technical Report ITL-94-5	Procedure for Static Analysis of Gravity Dams Including Foundation Effects Using the Finite Element Method - Phase 1 B	Jul 1994
Instruction Report ITL-94-5	User's Guide: Computer Program for Winkler Soil-Structure Interaction Analysis of Sheet-Pile Walls (CWALSSI)	Nov 1994

(Continued)

REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

	Title	Date
Instruction Report ITL-94-6	User's Guide: Computer Program for Analysis of Beam-Column Structures with Nonlinear Supports (CBEAMC)	Nov 1994
Instruction Report ITL-94-7	User's Guide to CTWALL - A Microcomputer Program for the Analysis of Retaining and Flood Walls	Dec 1994
Contract Report ITL-95-1	Comparison of Barge Impact Experimental and Finite Element Results for the Lower Miter Gate of Lock and Dam 26	Jun 1995
Technical Report ITL-95-5	Soil-Structure Interaction Parameters for Structured/Cemented Silts	Aug 1995
Instruction Report ITL-95-1	User's Guide: Computer Program for the Design and Investigation of Horizontally Framed Miter Gates Using the Load and Resistance Factor Criteria (CMITER-LRFD)	Aug 1995
Technical Report ITL-95-8	Constitutive Modeling of Concrete for Massive Concrete Structures, A Simplified Overview	Sep 1995
Instruction Report ITL-96-1	User's Guide: Computer Program for Two-Dimensional Dynamic Analysis of U-Frame or W-Frame Structures (CDWFRM)	Jun 1996
Instruction Report ITL-96-2	Computer-Aided Structural Modeling (CASM), Version 6.00 Report 1: Tutorial Guide Report 2: User's Guide Report 3: Scheme A Report 4: Scheme B Report 5: Scheme C	Jun 1996
Technical Report ITL-96-8	Hyperbolic Stress-Strain Parameters for Structured/Cemented Silts	Aug 1996
Instruction Report ITL-96-3	User's Guide: Computer Program for the Design and Investigation of Horizontally Framed Miter Gates Using the Load and Resistance Factor Criteria (CMITERW-LRFD) Windows Version	Sep 1996
Instruction Report ITL-97-1	User's Guide: Computer Aided Inspection Forms for Hydraulic Steel Structures (CAIF-HSS), Windows Version	Sep 1996
Instruction Report ITL-97-2	User's Guide: Arch Dam Stress Analysis System (ADSAS)	Aug 1996
Instruction Report ITL-98-1	User's Guide for the Three-Dimensional Stability Analysis/Design (3DSAD) Program	Sep 1998
Technical Report ITL-98-4	Investigation of At-Rest Soil Pressures due to Irregular Sloping Soil Surfaces and CSOILP User's Guide	Sep 1998
Technical Report ITL-98-5	The Shear Ring Method and the Program Ring Wall	Sep 1998
Technical Report ITL-98-6	Reliability and Stability Assessment of Concrete Gravity Structures (RCSLIDE): Theoretical Manual	Dec 1998

(Continued)

REPORTS PUBLISHED UNDER THE COMPUTER-AIDED STRUCTURAL ENGINEERING (CASE) PROJECT

(Concluded)

	Title	Page
Technical Report ITL-99-1	Development of an Improved Numerical Model for Concrete-to-Soil Interfaces in Soil-Structure Interaction Analyses Report 1: Preliminary Study Report 2: Final Study	Jan 1999 Aug 2000
Technical Report ITL-99-5	River Replacement Analysis	Dec 1999
ERDC/ITL TR-00-1	Evaluation and Comparison of Stability Analysis and Uplift Criteria for Concrete Gravity Dams by Three Federal Agencies	Jan 2000
ERDC/ITL TR-00-2	Reliability and Stability Assessment of Concrete Gravity Structures (RCSLIDE): User's Guide	Jul 2000
ERDC/ITL TR-00-5	Theoretical Manual for Pile Foundations	Nov 2000
ERDC/ITL TR-01-2	SOILSTRUCT-ALPHA for Personal Computers Report 1: Visual Modeler	Dec 2001